

Internship Report

PFE

Benjamin Martinet

Development of a Finite Element Model for the study of the contact between a cam and a roller applied in a real conjugated cam mechanism.



UNIVERSITAT POLITÈCNICA
DE CATALUNYA



Ecole Des Mines de Douai

From April, the 12th of 2012
To September, the 24th of 2012





Acknowledgements

I would like to thank Pau Català Calderón, Industrial Engineer and PhD student at the UPC for his welcome, his help and advices on the work, and also for his friendship.

Maria Antonia De los Santos, my tutor, for allowing me to do my internship at the ETSEIB, and also for her availability during the whole internship.

Jacques Boonaert, Head of the Productic Option at the Ecole des Mines de Douai, for having accepted my internship.

The PhD students From the Department of Mechanical Engineering of the ETSEIB, Ayoub, Bruno and Matteo for the time we shared.

The people I have met in Barcelona and the people who have visited me for the time we spent together.





Summary

Acknowledgements.....	3
Summary	5
Abstract.....	7
Résumé.....	9
Presentation of the host environment.....	11
I. Barcelona	11
II. Universitat Politècnica de Catalunya (UPC)	12
III. Escola Tècnica Superior d'Enginyeria Industrial de Barcelona (ETSEIB)	13
IV. Proposed Mission	13
Work done.....	15
I. Cam mechanisms	16
II. Hertz Contact theory	17
III. CAD model	21
IV. FEM model of the conjugated cams system.....	23
V. Misalignment comparison	28
VI. Difficulties.....	32
Personal Impressions	33
Conclusion	35
Bibliography	37
Figures	39
Tables	41
Annex	43
Annex 1: Principal Cam mechanical drawing.....	44
Annex 2: Conjugated Cam mechanical drawing.....	45
Annex 3: Roller Description - INA FAG	46
Annex 4: APDL program for High pressure case	47
Annex 5: Description of elements used in the model.....	49





Abstract

At the Ecole des Mines de Douai, the internships in companies are an important part of the students' schooling. Students have to do an internship in the company of their choice at the end of each year in order to get experience in a professional context. For the last six months, I have done my last internship (Final Year Project) in the Department of Mechanical Engineering in the Escola Tècnica Superior d'Enginyeria Industrial de Barcelona (ETSEIB) where I have been working in the scope of a Thesis realized by PhD candidate, Pau Català Calderón. The main subject of my work has been to develop a Finite Element Model to study the contact between the cam and the roller applied in a real conjugated cam mechanism. In order to realize the study, firstly a CAD model has been realized. The results obtained with the FEM models agree with the analytical results obtained via the Hertz Contact Theory. Also an additional study that compares two different types of rollers in case of a misalignment between the axis of rotation of the cam and the roller has been realized. I had to get familiar with new concepts, such as cam mechanism, the Hertz Theory and finite element method (FEM). Most of all, I had to learn to use the FEM software ANSYS v13 which has been in the center of my work here. Globally, this internship has confronted me to a research position, which has been very interesting. Besides, this internship has been for me the occasion to discover a new city and its culture, Barcelona. It was for me the first time living abroad, and it has been for sure a great experience.

Keywords

- | | |
|------------------|----------------------|
| -Research | -Cam Mechanism |
| -Finite Elements | - Hertz Theory |
| - ANSYS | - Final Year Project |



Résumé

A l'Ecole des Mines de Douai, les stages en entreprises occupent une partie importante du cursus scolaire des étudiants. Les élèves ingénieur doivent effectuer un stage dans l'entreprise de leur choix à la fin de chaque année dans le but d'obtenir de l'expérience dans un contexte professionnel. Durant l'été 2012, j'ai réalisé mon dernier stage (Projet de Fin d'Etude) dans le Département d'Ingénierie Mécanique de l'ETSEIB (Escola Tècnica Superior d'Enginyeria Industrial de Barcelona) où j'ai travaillé sur un projet de recherche dans le cadre d'une thèse réalisée par le doctorant, Pau Català Calderón. Le sujet principal de mon travail fut de développer un modèle utilisant la méthode des éléments finis afin d'étudier le contact entre une came et son suiveur appliqué au cas réel d'un mécanisme à cames conjuguées. Une étude additionnelle qui compare deux types de suiveurs dans un problème de défaut d'alignement des axes de rotation de la came et de son suiveur a également été réalisée. Afin d'assurer le bon déroulement de mon travail, j'ai dû me familiariser avec les concepts impliqués tels que: les mécanismes à came, la Théorie des Contacts de Hertz, ainsi que la méthode des éléments finis. Par dessus tout, j'ai dû apprendre à utiliser le logiciel d'éléments finis ANSYS v13 qui a été au centre de mon travail. Globalement ce stage m'a permis d'être confronté à une poste lié à la recherche, ce qui fut très intéressant. D'autre part, ce stage a été pour moi l'occasion de découvrir une nouvelle ville et sa culture, Barcelone. C'était aussi pour moi la première occasion de vivre à l'étranger, et ce fut bien sur une excellente expérience.

Mots clés

- | | |
|------------------|-------------------------|
| -Recherche | - Mécanismes à came |
| - Eléments finis | - Théorie de Hertz |
| - ANSYS v13 | - Projet de Fin d'Etude |



Presentation of the host environment

I. Barcelona

For the whole length of my internship, Barcelona (Figure 1.1 and 1.2) had been the city I was living in.



Fig 1.1: Sightseeing of Barcelona (MNAC, Torre Agbar, Overview from the Parc Güell, Sagrada Família)

Barcelona, capital of Catalonia, is the second town of Spain in matter of population with more than 1,6 million inhabitants. Located on the Mediterranean coast (Figure 1.2), Barcelona is considered as a Global City. It is indeed a major cultural and economic center, being one of the world's leading destination and which has a big impact on multiple fields of activity all around the world (arts, commerce, fashion, entertainment,...). It is important to note that there are two official languages in Catalonia: Catalan and Spanish.



Fig 1.2: Map of Spain

II. Universitat Politècnica de Catalunya (UPC)

The Polytechnic University of Catalonia is a public institution, specialized in Engineering, Architecture and Science degrees counting with 25 schools located in the following cities: Barcelona, Castelldefels, Manresa, Sant Cugat del Vallès, Terrassa, Igualada, Vilanova i la Geltrú and Mataró (Figure 1.3).



Fig 1.3: Localisation of UPC schools

The UPC has about 30 000 students and 2 500 professors and researchers. Besides, it is the University of Spain with the most International PhD Students.

III. Escola Tècnica Superior d'Enginyeria Industrial de Barcelona (ETSEIB)



Fig 1.4: Nowadays building and localisation of the ETSEIB

The ETSEIB (Figure 1.4) was created in 1851 and is one of the oldest Engineering School which still is in function. It offers to its students to follow courses in numerous fields. Such as: Energetic, Mechanic, Automatic, Electronic, Computer Science, Materials, Logistics, Chemistry, Industrial Management, Construction and others.

The School opens its doors to more than 3000 students, 470 teachers and 240 members of the administrative staff, in a 47000m² building.

IV. Proposed Mission

The internship has been realized in the Mechanical Engineering Department of the UPC which counts on several research groups. The enrolled research group has been the GREVTAM (Grup de Recerca en Vibracions i Teoria i Anàlisi de Maquines / *Group of Research in Vibrations and Theory and Analysis of Machines*). This research group divides its activities in basically two fields: the first one is dedicated to the vibrational aspect of mechanisms applied to rails machine (especially Barcelona's Subway). The second concern the synthesis and analysis of Machines and mechanisms like cam mechanisms, gears and bar mechanisms (Figure 1.5).

The study realized is in the scope of the line of investigation that deals with the synthesis and analysis of machines. This line of investigation is leaded by professors from the Department of Mechanical Engineering of the ETSEIB who have written many thesis and scientific journal papers related to the topic.



Fig 1.5: Use of cams, gears and bar mechanism

More concretely, the work done is directly related with a research done by the PhD student Pau Català Calderón and directed by Dr. Salvador Cardona and Dr. Maria Antonia De los Santos. The thesis deals with the influence that characterisation of the contact stiffness between the cam and the roller has in the dynamic response of the system when calculating deflections, forces and pressures contacts. The thesis proposes to use a non-linear contact stiffness based on the formulation of the general contact of the Hertz theory [1] instead of a linear stiffness based on the formulation for the cylindrical contact of the Hertz theory [2,3].

The study realized consists in developing a model with FEM for a real conjugate cam mechanisms (Figure 1.6) used on the production of wirehoods for cava bottles by industry Sàbat Lligats Metàlics S.L. The interest of this study for the company is that it can help providing a life time estimation for the elements of the cam mechanisms and plan new designs in order to improve life time.

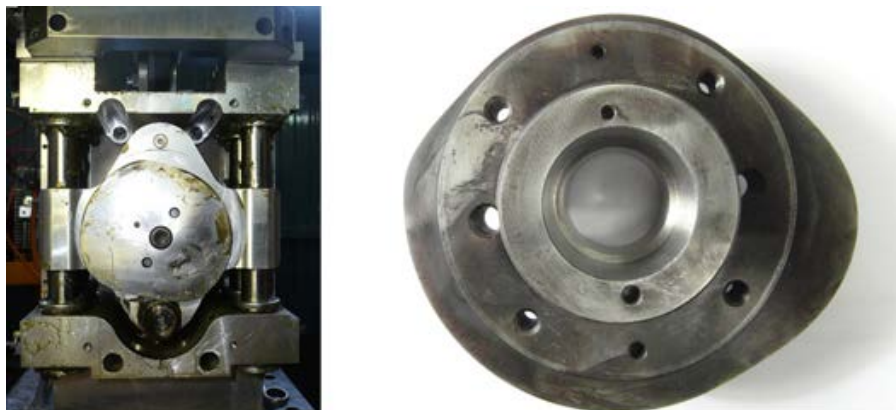


Fig 1.6: Pictures of the cams

Work done

The objectives of the work have been the followings:

1. Acquire fundamentals aspects regarding the analysis and synthesis of cam mechanisms.
2. Acquire fundamentals aspects regarding the Hertz contact theory.
3. Generate a CAD model for the real conjugate cam mechanism to export the geometry into a FEM program.
4. Develop with a FEM program a model to study the contact between the cam and the roller. The model obtained has to reproduce similar results than the ones predicted with the general contact formulation of the Hertz theory.
5. Compare the contact pressures obtained with cylindrical rollers and crowned rollers when there is a misalignment between the axis of rotation of the roller and the axis of rotation of the cam.

I. Cam mechanisms

The use of cam mechanism is to convert a rotational motion into an alternative linear or oscillating one. This certainly is the reason why this kind of mechanism is widely used in the shoe or textile industry or even in the every-day-life systems such as valve actuation in cars. The principal positive aspects of such mechanisms are: the high precision, the repeatability and its long life [4].

A cam mechanism is usually composed of two parts, the cam and the follower, as depicted in Figure 2.1:

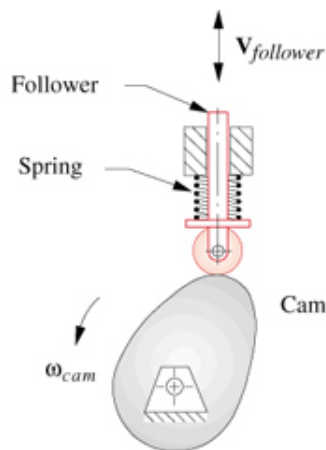


Fig 2.1: Cam mechanism [2]

There are several ways to classify cam mechanisms, depending on:

- Follower motion (translating / oscillating)
- Joint closure (force closure or form closure)
- Follower (flat faced, mushroom, roller)
- Cam (radial, axial or 3 dimensional)
- Motion constraints (Critical Extreme Position in which only some positions matters, or Critical Path Motion in which the whole path motion is important)
- Motion program (presence or not of dwells)

In Fig 2.1, a force closed mechanism is represented (the contact between the cam and the roller is assured with a spring) with a translating roller follower.

The real cam mechanism analysed (Figure 1.6) is a form closure joint, with roller follower, translational motion of the rollers and with a motion program with dwells and critical extreme positions. The two conjugated profile cams are axis linked and the two rollers are combined, imposing that the distance between the rollers is always the same. In order to avoid the follower to jump due to dynamical effects and manufacturing errors, the cam profiles are manufactured with an offset approximately of 125 μm in diameter.

Also the rollers mounted on the mechanism are crowned rollers which unlike basic rollers have not a flat outer surface (Figure 2.2).



Fig 2.2: Flat roller and crowned roller

II. Hertz Contact theory

The main part of the work has been to study the contact between the cam and the roller, that is, to evaluate the contact pressures depending on the cam position and the load applied. It is necessary to carry out studies about pressure contact to improve contact aptness and to have an idea of the life length of the mechanism.

The Hertz theory is applied to study the contact in many machine elements such as cams, gears, bearings. To apply the Hertz theory (1882) some conditions have to be checked. The load force must be normal to the tangent contact plane, the contact area must be small compared to the radii of the bodies in contact, the yield strength of the materials must not be exceeded and the system must be in equilibrium. The Hertz theory considers three contact formulations:

- Sphere against sphere.
- General contact (Figure 2.3a)
- Cylindrical contact (Figure 2.3b).

In a sphere against sphere and in the general contact situations, the nominal contact is a point. When two cylinders are in contact with the same axis orientation, the nominal contact is a line of zero width. Given these contact patches, any force applied would result in an infinite contact pressure. In this kind of contact the materials deflect and therefore create an area of contact.

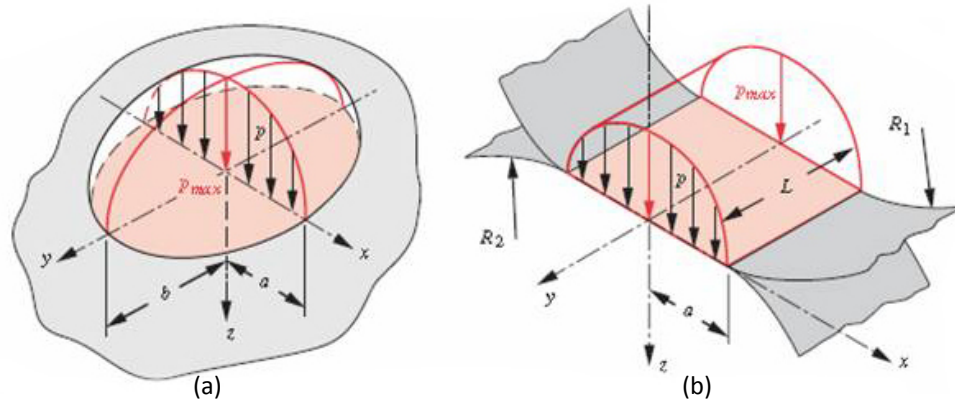


Fig. 2.3: Pressure distribution for general contact (a) and cylindrical contact (b). [2]

For the analysed conjugated cam mechanism that mounts crowned rollers, it is recommended the usage of the formulation of the general contact.

In this case, to define the geometry of each solid in contact (cam and roller), two radii of curvature are required (Figure 2.4). For the roller, the maximum radius R_1 is 500 mm and the minimum radius R_1' is 15 mm. For the cams, the maximum radius R_2 is considered infinite due to its planar shape along the axis direction and the minimum radius varies along the cam profile (Figure 2.10).

The different variables that come in consideration when studying a Hertz contact are summarized in table 2.1:

R_1	First radius of curvature, solid 1	m
R_1'	Second radius of curvature, solid 1	m
R_2	First radius of curvature, solid 2	m
R_2'	Second radius of curvature, solid 2	m
θ	Angle between the planes containing R_1 and R_2	
E_1	Young modulus for solid 1	Pa
E_2	Young modulus for solid 2	Pa
ν_1	poisson ratio for solid1	/
ν_2	Poisson ratio for solid2	/
a	Half width of contact patch major axis	m
b	Half width of contact patch minor axis	m
F	Force	N
P	Pressure	Pa

Table 2.1: List of variables.

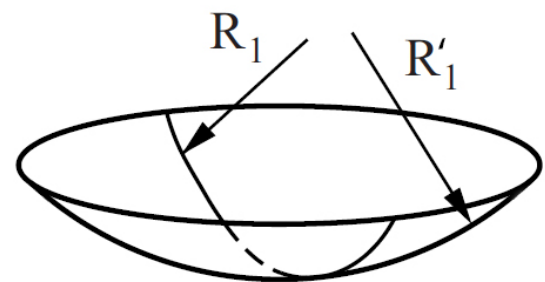


Fig. 2.4: Representation of radii.

When the two bodies in contact are made of any general curvature, the resulting contact patch is an ellipse and the pressure distribution consist in a semi-ellipsoid as you can see in Fig 2.3. We have a maximum pressure P_{\max} at the center and the pressure is null at the edge. The load F is equal to the volume of the semi-ellipsoid:

$$F = \frac{2}{3} \pi ab P_{\max} \quad \text{or} \quad P_{\max} = \frac{3}{2} \frac{F}{\pi ab} \quad (1)$$

The major axis a and the minor axis b of the semi-ellipsoid are calculated with the following formulas:

$$a = k_a \sqrt[3]{\frac{3}{4} F \frac{m_1 + m_2}{A}} \quad b = k_b \sqrt[3]{\frac{3}{4} F \frac{m_1 + m_2}{A}} \quad (2)$$

Where m_1 , m_2 , A , k_1 and k_2 are obtained with the following equations:

$$m_1 = \frac{1 - \nu_1^2}{E_1} \quad m_2 = \frac{1 - \nu_2^2}{E_2} \quad (3)$$

$$A = \frac{1}{2} \left(\frac{1}{R_1} + \frac{1}{R'_1} + \frac{1}{R_2} + \frac{1}{R'_2} \right) \quad (4)$$

In order to calculate k_a and k_b , B and ϕ are needed:

$$B = \frac{1}{2} \left[\left(\frac{1}{R_1} - \frac{1}{R'_1} \right)^2 + \left(\frac{1}{R_2} - \frac{1}{R'_2} \right)^2 + 2 \left(\frac{1}{R_1} - \frac{1}{R'_1} \right) \left(\frac{1}{R_2} - \frac{1}{R'_2} \right) \cos 2\theta \right] \quad (5)$$

$$\phi = \cos^{-1} \left(\frac{B}{A} \right) \quad (6)$$

Using ϕ , k_a and k_b are determined using the following table:

ϕ [°]	0	10	20	30	35	40	45	50	55	60	65	70	75	80	85	90
k_a	∞	6,612	3,778	2,731	2,397	2,136	1,926	1,754	1,611	1,486	1,378	1,284	1,202	1,128	1,061	1
k_b	0	0,319	0,408	0,493	0,53	0,567	0,604	0,641	0,678	0,717	0,759	0,802	0,846	0,893	0,944	1

Table. 2.2: Factors in use for the calculation [2]

With (1) and knowing F , it gives P_{max} , and then with (8) the pressure distribution is calculable:

$$P = P_{max} \sqrt{1 - \frac{x^2}{a^2} + \frac{y^2}{b^2}} \quad (7)$$

Also, the force contact depends on the common deformation δ between the two solids and the contact stiffness k_H .

$$Fc = k_H \delta^{3/2} \quad (8)$$

In the studied conjugated cam mechanism, the profiles are manufactured with an extra offset of 125 μ m. But considering the manufacturing errors of the cams and the rollers, it gives an estimated approach between cam's and roller's center of 25 μ m. Solving the dynamic equations of the system of three degrees of freedom done by Català et al. [5] gives an approach between each the cam and the roller of 6,5 μ m. Imposing the approach and assuming that the stiffness is between 54 GN/m^{3/2} and 55 GN/m^{3/2}, the preload applied to the mechanism is estimated to 1000 N.

III. CAD model

Although Ansys is a complete FEM method software which allow 3D Design, Solidworks provides way more handy possibilities to generate the profile of the cam(Fig 2.5). Every cam profile was generated with 3600 points given by the company.

The resulting CAD model has been used to compare theoretical dimensions of the conjugated cams with actual ones, using a Coordinates Measure Machine (CMM, Figure 2.7) and to evaluate the manufacturing errors which has been useful for Sàbat Lligats Metal·lics because it had not been done before.

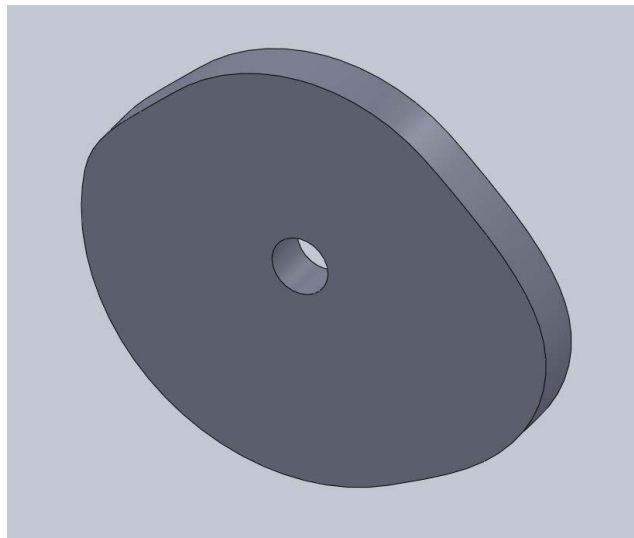


Fig 2.5: Model of the cam that will be used in the study

By concern of a better understanding of the system, both conjugated cams were modeled and the rollers and the screw details were added, according to Sàbat Lligats Metal·lics information (Annexes 1 and 2) (Fig 2.6).

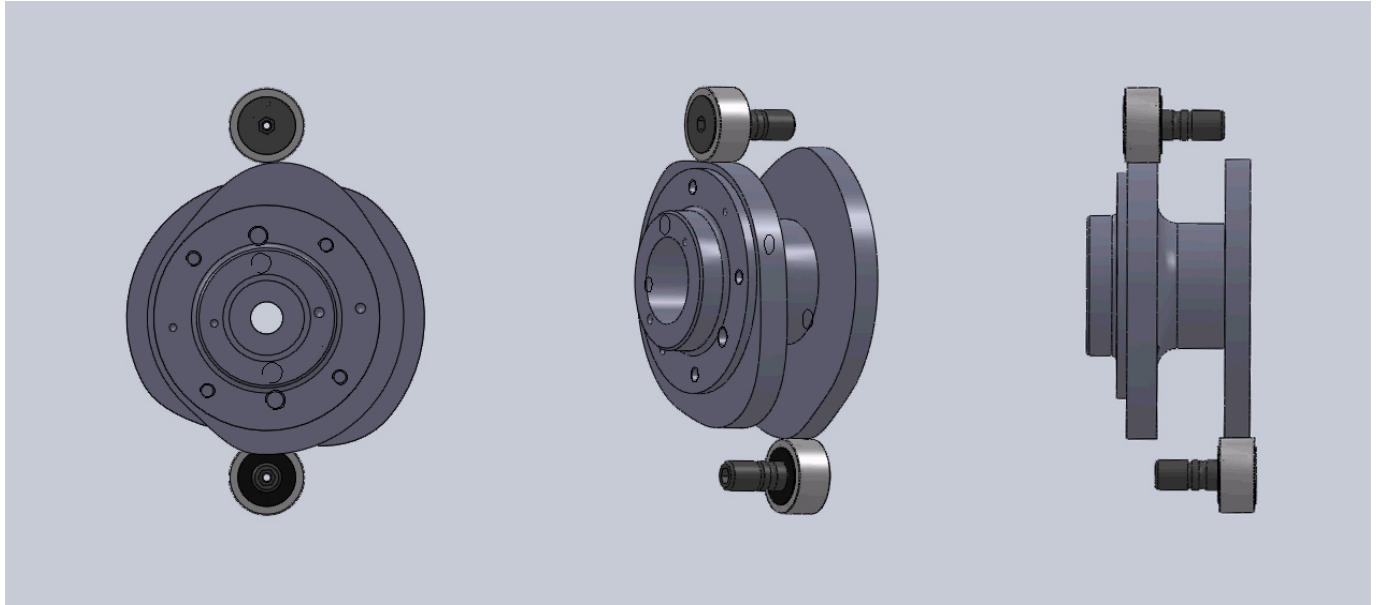


Fig. 2.6: Different views of the conjugated cams

The CAD model has been useful in order to export the complex geometry of the cam system to ANSYS.



Fig 2.7: Coordinate Measure Machine.

IV. FEM model of the conjugated cams system

The first attempts to generate a FEM model of the contact between the cam and roller were done with ANSYS Workbench but the results obtained were not accurate. After consulting bibliography related with the FEM applied in the contact cases [6] the next attempts were tried with ANSYS APDL. Although the use of ANSYS Workbench is easier thanks to a user-friendly interface, ANSYS APDL allows the importation of the CAD model generated with Solidworks and offers more possibilities on the configuration of parameters. Several tutorials were followed [6] [7], and finally the use of Ansys Parametric Design Language (APDL) revealed to be the most efficient solution. Indeed, with APDL, models are generated from text file (Annex 4), therefore the modification of models is very quick

The first attempts to learn the usage of ANSYS APDL and obtain results with a FEM model that reproduces the Hertz contact results have been done with simple geometries. That is sphere and sphere contact (Figure 2.8 and 2.9), sphere and plane or cylinder and cylinder.

When the solution converge, it is possible to display contact pressure evolution on the model. Also, the list of contact nodes with their pressure is available, by combining this list with the list of nodes and coordinates of the whole model appears the possibility to plot the contact patch with a calculation software (SciLab).

Sphere against Sphere example:

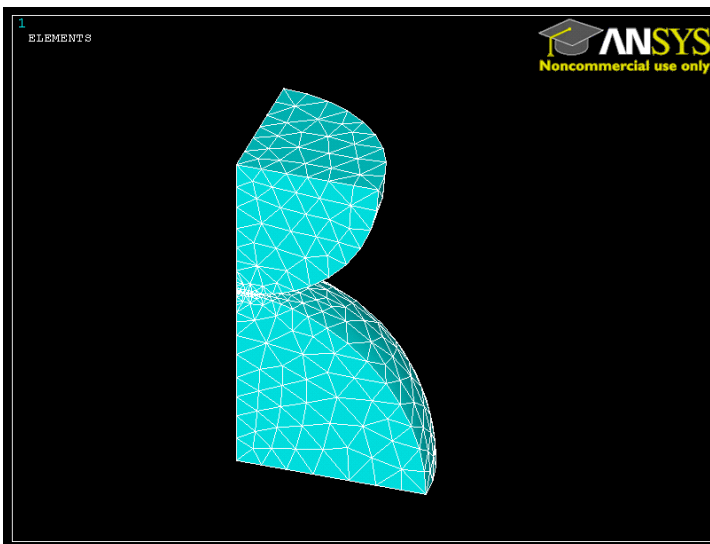


Fig2.8: Sphere/Sphere model.

Parameters:

Lower Sphere Radius: 40 mm

Upper Sphere Radius: 30 mm

Young Modulus: 210E9 Pa

Poisson's Ratio: 0,3

Interference =50 μ m

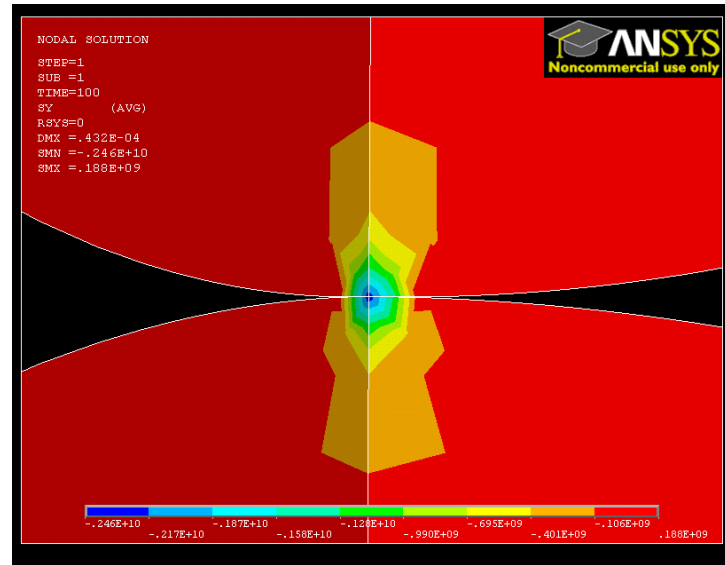


Fig2.9: Plotted Y component of Stress.

The results obtained with ANSYS and with the Hertz contact formulation for the case sphere to sphere (which is a particular case of the general contact formulation) are resumed in Table 2.3.

	Ansys	Theory	Relative diff
Pressure [MPa]	1947,4	2056	5,28%
$\bar{\delta}$ [μm]	13,34	13,49	1,11%

Table 2.3: Results table, Spheres cases.

The differences between ANSYS results and the ones obtained with the Hertz theory are sufficiently small. As results of the study and with the help of Ansys Contact Guide [8] of this case, the interest of using the elements CONTACT 174, TARGE170 and SOLID187 (ANNEX 5) came out.

Because the cams have a different radius of curvature (r_c) on each point of the profile (Figure 2.10), for each contact point a different contact pressure is expected. In Figure 2.11 are depicted the contact pressure obtained with the formulation of the general contact of the Hertz theory.

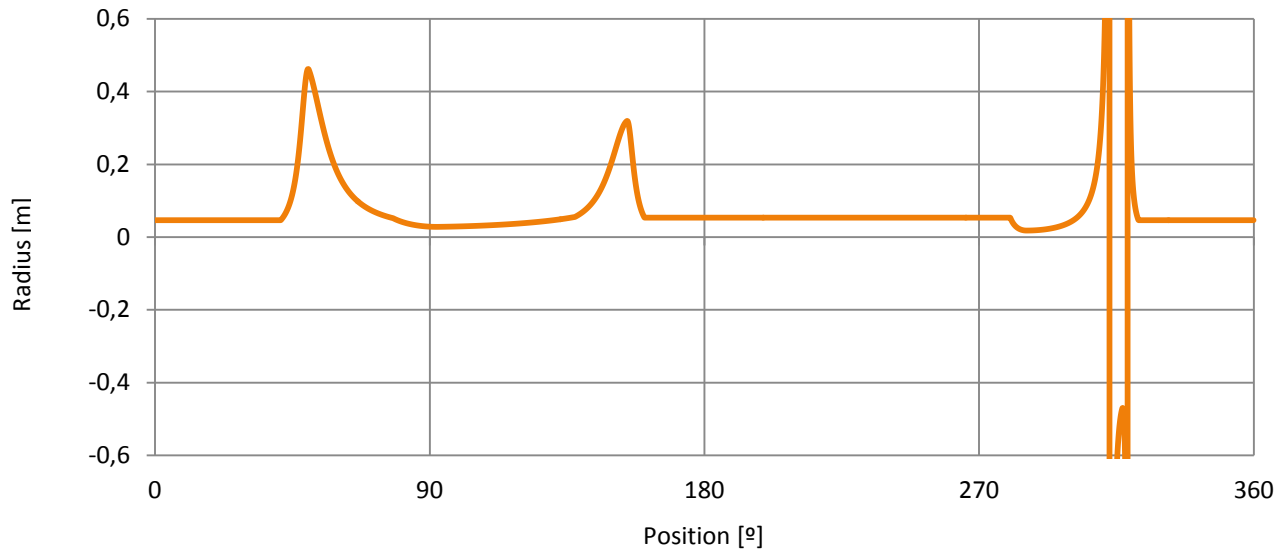


Fig. 2.10: Evolution of radius depending on the position

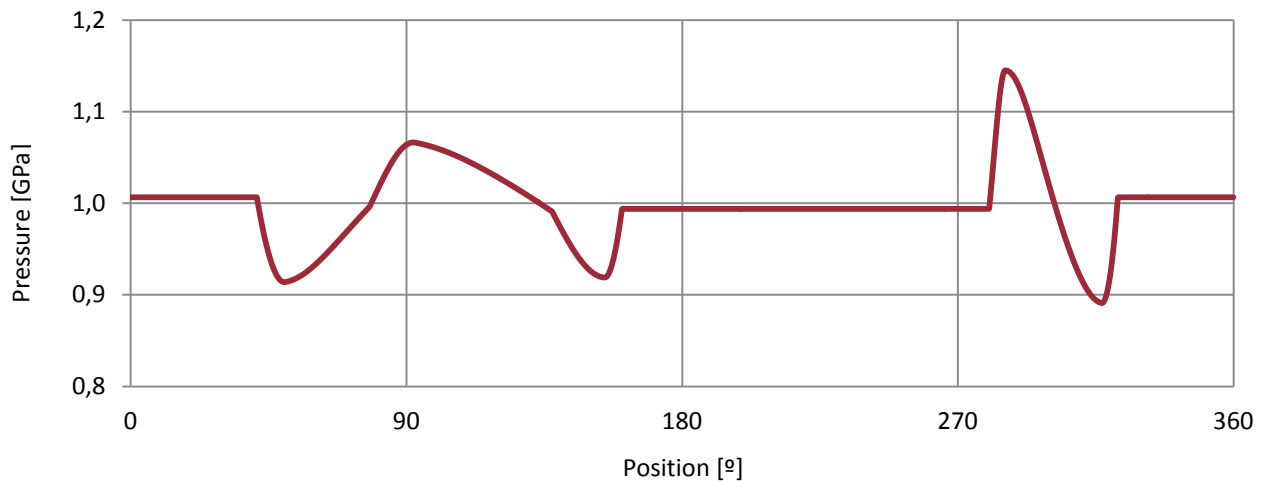


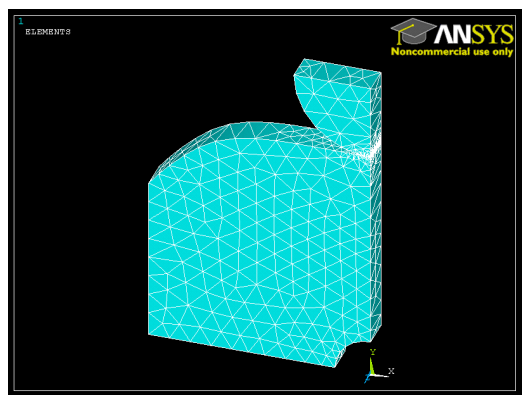
Fig. 2.11: Evolution of pressure depending on the position

Three different contact points were decided to be studied. The first cam point is where the cam has a minimal convex radius of curvature of 19 mm, this is the point where the maximum contact pressure appears and the point where the surface fatigue is the most important. The second cam point is where the cam has the minimal concave radius, approximately -460 mm, where minimum contact pressure is expected. The last studied point of the cam is where the pressure angle is the highest. Obtaining the results in any other point should be equivalent thanks to the program (Annex 4).

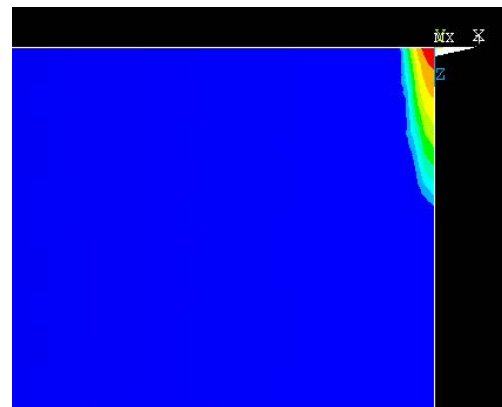
For the three cases, the parameters are the following:

The roller model is made after the KR30PP track roller by INA FAG (Description in Annex 3)
 Material is considered as structural steel (Young Modulus: 206,8 GPa, Poisson ratio: 0,28).
 The applied force is F and has a value of 1000N.
 The elements used are the following: TARGE170 and CONTA174.

In the following figures are represented the FEM model where the roller is in contact with the point of the cam that has the minimum radius (Figure 2.12 a), the point of the cam that has the maximum concave radius (Figure 2.13 a) and the point of the cam where the transmission of the load is maximum (Figure 2.14 a). In order the calculation to be faster and thanks to the symmetry of the model, only quarter models have been realized.

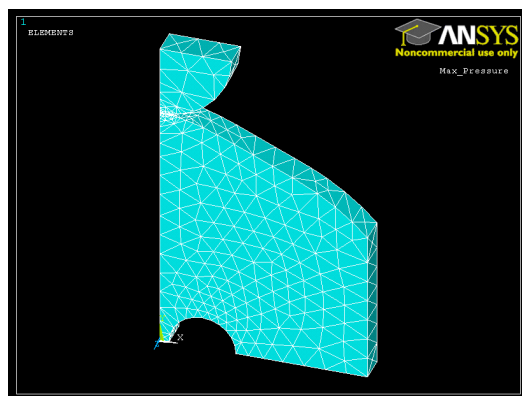


(a)

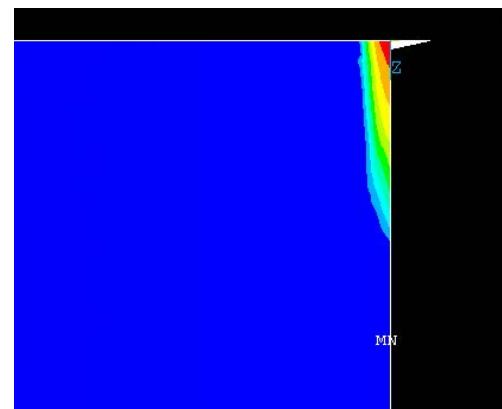


(b)

Fig 2.12: Model and repartition of pressure for the lowest pressure case



(a)



(b)

Fig 2.13: Model and repartition of pressure for the highest pressure case

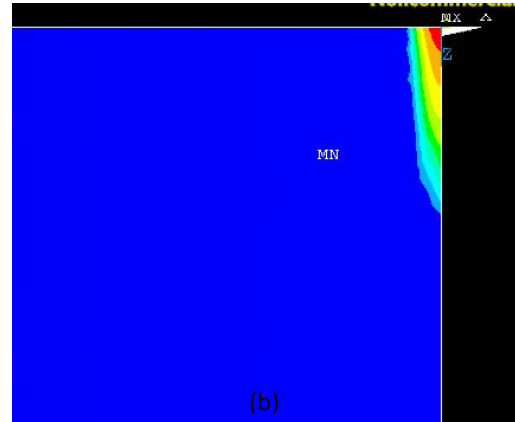
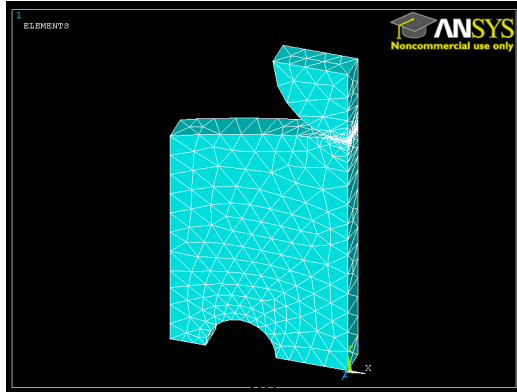


Fig2.14: Model and repartition of pressure for the highest pressure angle

As result, in figures 2.12, 2.13 and 2.14 (b), the contact area is noticeable and as expected, the pressure is maximum at the center.

Results:

		Ansys	Theory	Relative diff
Low. Pressure $r_c=464,9\text{mm}$	Pressure [MPa]	890,92	867,9	2,65%
	δ [μm]	8,3	6,89	20,46%
High. Pressure $r_c=19,15\text{mm}$	Pressure[MPa]	1148,84	1114	3,13%
	δ [μm]	9,1	7,26	25,34%
Max. Force $r_c=53,51\text{mm}$	Pressure [MPa]	1006,79	995,5	1,13%
	δ [μm]	9,48	7,06	34,28%

Table 2.4: Repartition of Pressure (Top view of the contact surface)

It can also be noted that the displacement measures are inaccurate and that a finer mesh does not have repercussion on it. This could be due to initial settings in the model. Indeed, ANSYS needs and interference between the solids in contact in order to recognize the contact. The interference parameter could have a repercussion on the final result.

Comparing these results with the work of Nagaraj Nayak et al.[9] who obtain a difference of approximately 25% between expectations and results, these results are enough accurate, especially regarding pressures. The displacement measures (common deformation of the cam and the roller) are much more inaccurate and that a finer mesh does not have repercussion on it.

V. Misalignment comparison

There are two types of rollers followers: ones with cylindrical outer surface and ones with crowned outer surface. Although crowned rollers a priori can support lower contact pressures, their advantages are that they assure an almost a constant contact pressure in a bigger range of misalignment (Figure 2.15) between the cam and the roller axis of rotation. Also, Sàbat Lligats Metal·lics' experience proves that the life time obtained with crowned rollers is bigger than with cylindrical ones. Di et al. [10] also arrive at the conclusion that lower contact pressures are obtained with crowned rollers in a diesel engine.



Fig 2.15: Tilting of a roller

In this chapter a comparison between cylindrical rollers and crowned rollers is realized with ANSYS in order to reproduce the results obtained by the INA-FAG roller manufacturer [INA-FAG website].

For this study, the parameters are the following:

The model is made after the NUKR80 track roller by INA FAG.

Material is considered as structural steel (Young Modulus: 206,8 GPa, Poisson ratio: 0,28).

The applied force is F and has a value of 13800N.

The elements used are the following: TARGE170 and CONTA174.

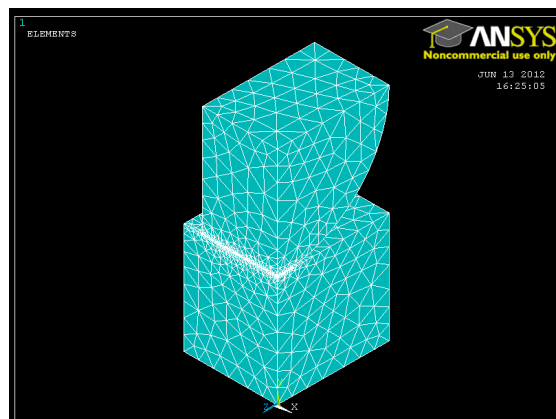


Fig 2.16: Model (Differences between tilted/crowned model is not noticeable).

Cylindrical Roller

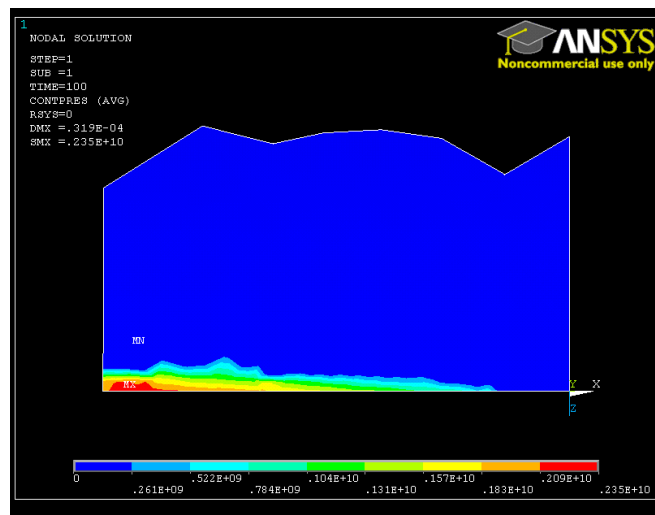
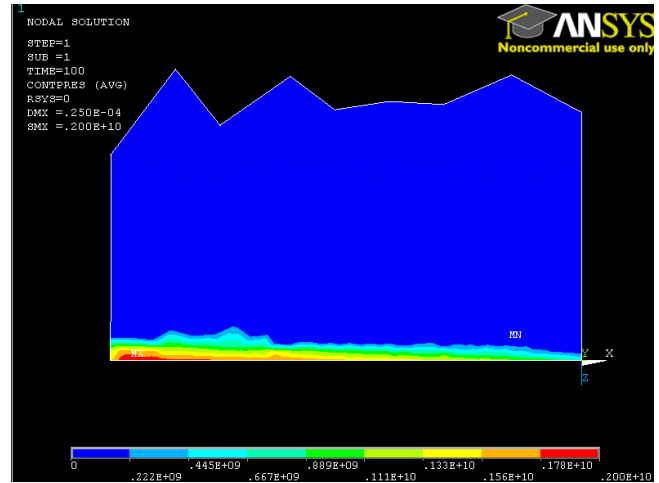
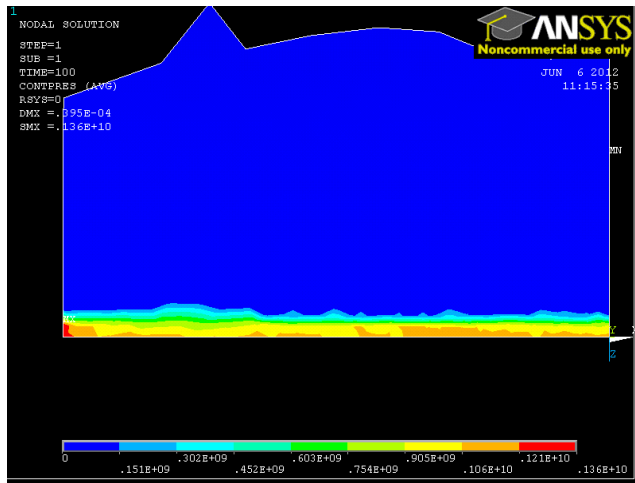


Fig 2.17: Evolution of the repartition of pressure (Top view of the contact surface), Cylindrical profile.

Tilt [mrad]	0	1	2
Pressure [GPa]	0,885	1,338	1,636

Table 2.5: Evolution of the pressure, Cylindrical profile.

As the roller gets tilted, the contact area decreases implying a rise of the pressure.

Crowned roller

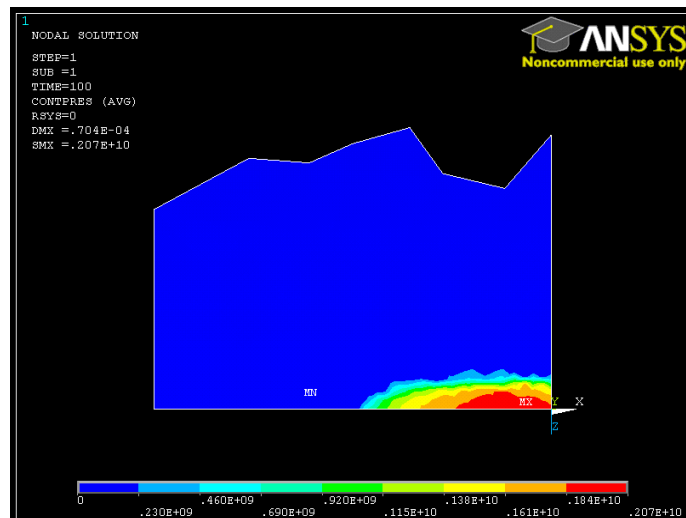
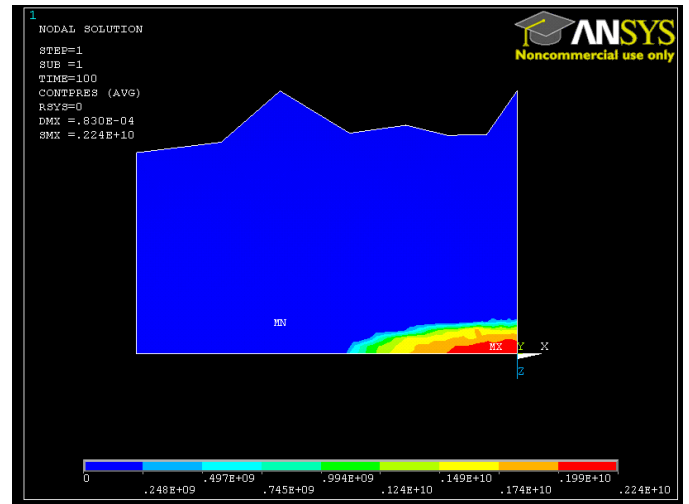
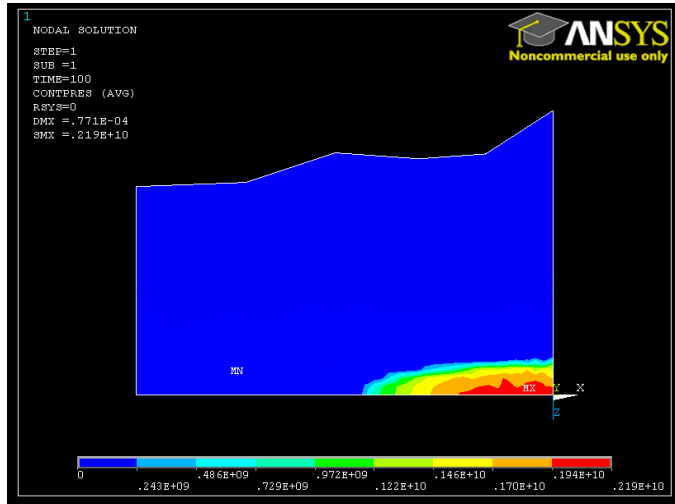


Fig2.18: Evolution of the repartition of pressure (Top view of the contact surface), Crowned profile

Tilt [mrad]	0	1	2
Pressure [GPa]	1,383	1,367	1,273

Table 2.6: Evolution of the pressure, Crowned profile.

Results

Tilt [mrad]	0	1	2
Cylindrical	0,885	1,338	1,636
Crowned	1,383	1,367	1,273

Table 2.7: Comparison of the results

Those results confirm the utility of a crowned profile of the roller to deal with a very probable inclination problem (one or two milli-radians angles). The pressure on a crowned profile will tend to be constant regardless of the inclination thanks to its curved outer surface, as we could expect it. There is no analytical formulation to calculate the effect of the tilt between the cam and the roller. The FEM models generated are a good tool to study this effect.

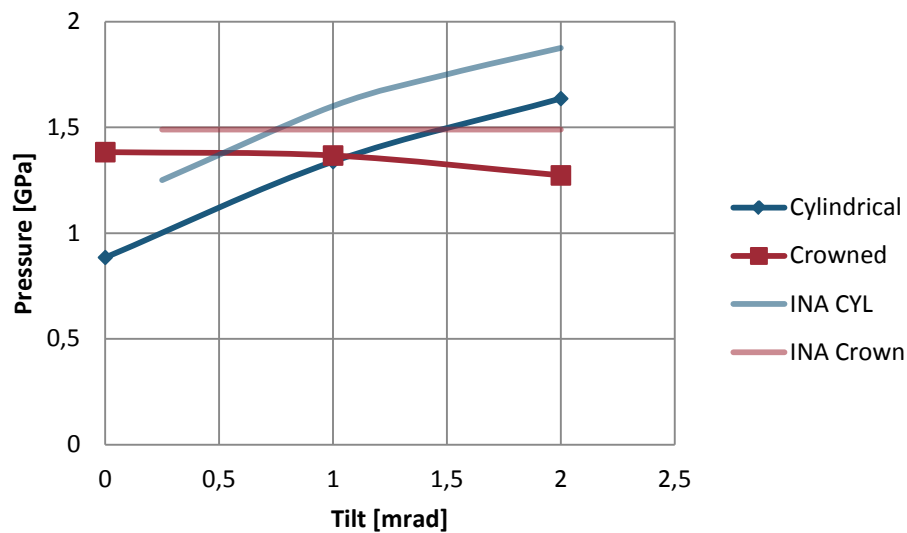


Fig2.19: Comparison of the evolution of Pressure between cylindrical et crowned profiles

Globally, the results match with the ones of the manufacturer and the idea is well transcribed with ANSYS, though there is a slight difference between results even with a fine mesh.

VI. Difficulties

Ansys V13 is a very complete and complex software, and it took me time to get used to it. I would even say that I finally know how to use a small part of what it proposes.

The beginning with Ansys Workbench was pretty confusing, I tried several cases of Hertz contact, but none of them were correct. We took the decision to move to ANSYS APDL, knowing that it would take some more time but also that the results would be more reliable.

To calibrate the model I had to learn about the different elements in use in Ansys 13, and to select in a large list the one that would suit the study. I had to refer to the Ansys Contact Guide which is a book of approximately 150 pages about contact in Ansys, this can give a idea of the complexity of the software.

I would also say that as I am not a mechanical engineering student, it could appear difficult to confront a mechanical problem like this one. But given the time I had and the interest I have for mechanics, I have been able to learn the things I needed to carry out the study.



Personal Impressions

Like the previous internships I have done, I think this one has been a great experience for my future. In order to explain this, I will talk about internships in general, and then make the point with my professional project.

In my opinion, internships are one of the most important things in the engineer schooling. Indeed, they contain a lot of things the student engineer will need to start in his career; for example, taking contact with companies, writing cover letters and resume, and going through interviews. But most of all, it helps us to get use to the company environment, to interact with its different actors. Plus, it makes us travel, discovers new regions or countries, and it develop a certain kind of autonomy. Globally, I would say that Internship develop our capacity to adapt and complete the courses we follow at the Ecole des Mines de Douai with practical application.

When I had to send my applications for this internship, I have chosen to send it to some universities because I wanted to have another experience; I wanted to have a look at research.

Now I would say that I enjoyed this experience, because it allowed me to learn a lot of things. Staying in Barcelona has permitted me to learn about Spanish and Catalan culture, and if my improvement in Spanish is not so big, I learned enough to interact with Barcelonans and to talk with people I have met there.

I also learned a lot of things about cam mechanisms, about the different kinds that exist, how they work, why they are useful, their design procedure. I can now imagine what it involves to be a specialist in cam mechanisms. But I have not only learned cam mechanisms, Pau Català, the PhD student I have been working with, gives classes to student about Machines Theory, so he proposed to show me some of the mechanisms they had in the department, so that I have been able to learn about engines, car gearbox, gears, cams and bar mechanisms. He has also helped me a lot in the understanding of the Hertz theory.

I am now able to use the Finite Element Method software ANSYS, I have also spent some time in SciLab, and I have reinforced my knowledge in mechanics; I am sure this will be a plus for me in my professional future.

Professionally speaking, this internship has reinforced my interest for a work placement in a design office/research department as I feel that it would suit better my professional expectations.

I am pleased with what I have done here, and what I have liked the most is the possibility to go through with what you are working and to have the necessary time to carry things out.

From the many things I have learnt, I will for sure remember that it takes time to handle a FEM software. The numbers of parameters is impressive, and the documentation is very large.



Conclusions

A CAD model of the real conjugate cam mechanism has been generated with Solidworks. This CAD model has been used to obtain manufacturing errors for the real profiles of the cam with a CMM.

Also a FEM model of the cam mechanism has been generated with ANSYS APDL that reproduces enough accurately the analytical results obtained via the Hertz formulation for the general contact regarding contact pressures. Although just three contact cases have been studied, the usage of a programmatic allows to the PhD candidate Pau Català to reproduce easily the results in any point of the cam profile.

Also the FEM has proved that it is a good method to obtain results when exists a misalignment between the cam and the roller axis.

The future tasks can consists in generating FEM models with a complete cam-roller mechanism and analysing the effects of tilting the axis of rotation on a real cam roller situation. An investigation on the reason why the values of deformation are not as good as the ones for pressure could also be interesting.



Bibliography

- [1] P. Català, S. Cardona, M.A. De los Santos, J. Martinez, *Influencia de la rigidez de contacto leva-palpador en la desviación de la ley de desplazamiento*, 2011.
- [2] M. Tounsi, F. Chaari, M. S. Abbes, T. Fakhfakh, M. Haddar, *Failure Analysis of a Cam-Follower System Affected by a Crack*, 2010.
- [3] E. Bouyer, S. Caro, D. Chablat, J. Angeles, *The Multiobjective Optimization of a Prismatic Drive*, 2007.
- [4] Robert L. Norton, *Cam Design and Manufacturing Handbook*, Industrial Press, 2002.
- [5] P. Català, S. Cardona, M.A. De los Santos, J.M. Veciana, *Influence of the contact stiffness between the cam and the roller in the dynamic behaviour of the train follower applied in a conjugate cam mechanism*, 2012.
- [6] Cornell University, *Ansys learning modules*, <https://confluence.cornell.edu/display/SIMULATION/ANSYS+Learning+Modules>.
- [7] University of Alberta, *Ansys tutorials*, <http://www.mece.ualberta.ca/tutorials/ansys/>
- [8] ANSYS, *Contact Technology Guide*, 2009
- [9] Nagaraj Nayak, P.A Lakshminarayanan, M.K. Gajendra Babu, A.D. Dani, *Predictions of cam follower wear in diesel engines*, 2005.
- [10] Tan Di and Ou Yang Guang-yao, *Analysis on Cam-roller Contact Stress and Optimization on Roller Figuration of Fuel-Injection System*, 2010.



Figures

Fig 1.1: Pictures of Barcelona (MNAC, Torre Agbar, Overview from the Parc Guëll, Sagrada Familia).....	11
Fig 1.2: Map of Spain.....	11
Fig 1.3: Localisation of UPC schools.....	12
Fig 1.4: Picture and localisation of the ETSEIB.....	13
Fig 1.5: Use of cams, gears and bar mechanism.....	14
Fig 1.6: Pictures of the cams.....	14
Fig 2.1: Cam mechanism.....	16
Fig 2.2: Flat roller and crowned roller.....	17
Fig 2.3: Pressure distribution for general contact (a) and cylindrical contact (b).....	18
Fig 2.4: Representation of radii.....	18
Fig 2.5: Model of the cam that will be used in the study.....	21
Fig 2.6: Different views of the conjugated cams.....	22
Fig 2.7: Coordinate Measure Machine.....	22
Fig 2.8: Sphere/Sphere model.....	23
Fig 2.9: Plotted Y component of Stress.....	24
Fig 2.10: Evolution of radius depending on the position.....	25
Fig 2.11: Evolution of pressure depending on the position.....	25
Fig 2.12: Model and repartition of pressure for the lowest pressure case.....	26
Fig 2.13: Model and repartition of pressure for the highest pressure case.....	26
Fig2.14: Model and repartition of pressure for the highest pressure angle.....	27
Fig 2.15: Tilting of a roller.....	28
Fig 2.16: Model (Differences between tilted/crowned model is not noticeable).	28
Fig 2.17: Evolution of the repartition of pressure (Top view of the contact surface), Cylindrical profile.....	29
Fig2.18: Evolution of the repartition of pressure (Top view of the contact surface), Crowned profile.....	30
Fig2.19: Comparison of the evolution of Pressure between cylindrical et crowned profiles.....	31





Tables

Table 2.1: List of variables.....	18
Table 2.2: Factors in use for the calculation.....	19
Table 2.3: Results table, Spheres cases.....	24
Table 2.4: Repartition of Pressure (Top view of the contact surface).....	27
Table 2.5: Evolution of the pressure, Cylindrical profile.....	29
Table 2.6: Evolution of the pressure, Crowned profile.....	30
Table 2.7: Comparison of the results.....	31





Annex

Annex 1: Principal Cam mechanical drawing

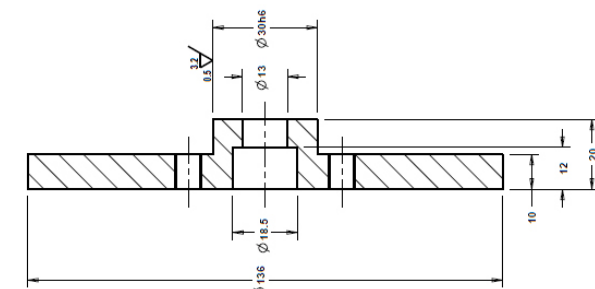
Annex 2: Conjugated Cam mechanical drawing

Annex 3: Roller Description - INA FAG

Annex 4: APDL program for High pressure case

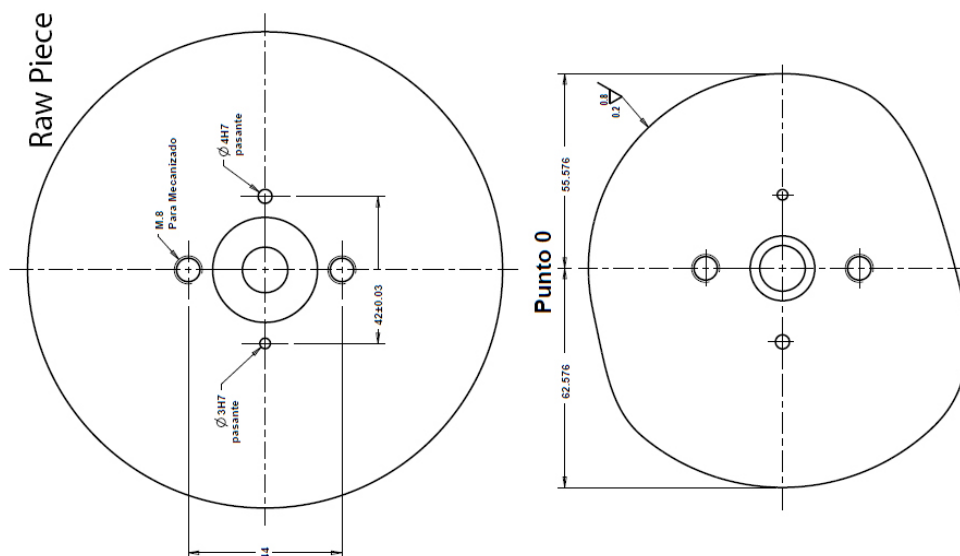
Annex 5: Description of elements used in the model

Annex 2: Conjugated Cam mechanical drawing



Dados Calcul:
 Diámetro Rod: 126 mm.
 Diámetro Rod: 30 mm.
 Precarrega 0.125 mm.
 Carrera leva 15.000 mm

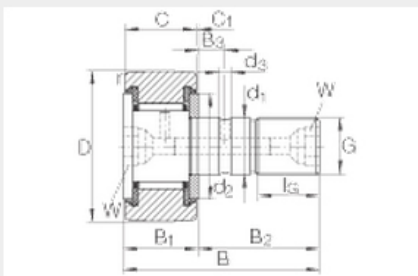
070-01907		Leva Exterior act. vert. doble torsion	
SABAT	Manuel Sabat	F-154	Cementado 60 63 Rc
	24/11/2009	Prof. Ø 30 precarga 0.05mm	Rectificado
	A.0		1.000
			1.000



Annex 3: Roller Description - INA FAG

Stud type track rollers KR30-PP

with axial guidance, plastic axial plain washers on both sides



D	30 mm	
d₁	12 mm	Tolerance: h7
B	40 mm	
B_{1 max}	15,2 mm	
B₂	25 mm	
B₃	6 mm	
C	14 mm	
C₁	0,6 mm	
d₂	23 mm	
d₃	3 mm	
G	M12x1,5	
l_G	13 mm	
r_{min}	0,6 mm	
W	6 mm	Nominal dimension for hexagonal socket
m	88 g	Mass
M_A	22 Nm	Nut tightening torque
C_{r w}	6800 N	Effective dynamic load rating as track roller (radial)
C_{0r w}	8600 N	Effective static load rating as track roller (radial)
C_{ur w}	1220 N	Fatigue limit load
n_{D G}	5500 1/min	Speed (with continuous operation and grease lubrication)
	NIPA1X4,5	Drive fit lubrication nipple Drive fit lubrication nipples are supplied loose. Only these lubrication nipples should be used.



Annex 4: APDL program for High pressure case

```
Finish
/clear

! parameters
rc=53.40313809
rr=15
i=5e-4
c=500-499.95009976
E=2.068e5
nu=0.28

! Import
/AUX15
IOPTN, IGES, SMOOTH
IOPTN, MERGE, YES
IOPTN, SOLID, YES
IOPTN, SMALL, YES
IOPTN, GTOLER, DEFA
IGESIN, 'maxpressure', 'IGS',
'Escritorio\Benjamin\Solidworks\pour_etude\redit'

/prep7

! Elements
ET, 1, SOLID187
ET, 2, TARGE170
ET, 3, CONTA174
Keyopt, 3, 10, 2

! Material
MP, EX, 1, E
MP, PRXY, 1, nu

! Keypoints
K, 51, 0, -i, 0,
K, 52, 0, -i+c, 6,
K, 53, 0, -i+rr, 6,
K, 54, 0, -i+rr, 0,
K, 55, 0, -i+500, 0,

! Lines
l, 51, 54
l, 54, 53
l, 53, 52
```

```
! Arc
LARC, 51, 52, 55, 500

! Area
al, 1, 2, 3, 4
! volume
vrotat, 1, , , , , 53, 54, 90

! mesh
smrsize, 1
vmesh, all

! Refining
nsl, s, node, , 1, 7, 6, ,
nsl, a, node, , 258, 260, 2, ,
nsl, a, node, , 191, 193, 2, ,
nsl, a, node, , 67, 75, 2, ,
nsl, a, node, , 369, 377, 2, ,
nrefine, all, , , 2

! selection of target elts
ASEL, s, area, , 8, 8, , 1
nsla, s, 1
nsl, r, loc, y, -2, 0.5
esel, all
TYPE, 2
ESURF,
allsel, all

! selection of conta elts
ASEL, s, area, , 4, 4, , 1
nsla, s, 1
nsl, r, loc, y, -i-0.5, -i+2
esel, all
type, 3
esurf
allsel, all

! selection of elts for coupling
ASEL, s, area, , 5, 5, , 1
nsla, s, 1
cp, 1, uy, all
allsel, all

! LOADS
```



```
!symmetry
ASEL,s,area,, 1, 2, 1, 0
ASEL,a,area,, 7, 9, 2, 0
da, all, symm
allsel, all
```

```
!DOF
ASEL, s, area,, 11, 11,, 0
da, all, uy, 0, 1
allsel, all
```

```
!force
fk, 54, fy, -250
```

```
Finish
```

```
/SOL
ANTYPE, 0
NLGEOM, 1
AUTOTS, 0
TIME, 100
SOLVE
```

```
!plot pressure
/post1
PLNSOL, cont, pres
esel, s, ename,, 174
prnsol, dof, uy
prnsol, cont,
```


Annex 5: Description of elements used in the model

SOLID187

3-D 10-Node Tetrahedral Structural Solid

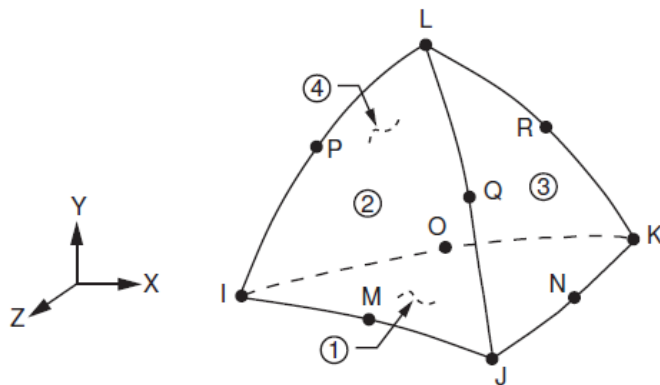
MP ME ST PR PRN DS DSS <> <> <> <> PP VT EME MFS
Product Restrictions

SOLID187 Element Description

SOLID187 element is a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes (such as those produced from various CAD/CAM systems).

The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyperelasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyperelastic materials. See [SOLID187](#) in the [Theory Reference for the Mechanical APDL and Mechanical Applications](#) for more details about this element.

Figure 1 SOLID187 Geometry





TARGE170

3-D Target Segment

MP ME ST PR PRN DS DSS <> EM <> <> PP <> EME MFS
Product Restrictions

TARGE170 Element Description

TARGE170 is used to represent various 3-D "target" surfaces for the associated contact elements ([CONTA173](#), [CONTA174](#), [CONTA175](#), [CONTA176](#), and [CONTA177](#)). The contact elements themselves overlay the solid, shell, or line elements describing the boundary of a deformable body and are potentially in contact with the target surface, defined by TARGE170. This target surface is discretized by a set of target segment elements (TARGE170) and is paired with its associated contact surface via a shared real constant set. You can impose any translational or rotational displacement, temperature, voltage, and magnetic potential on the target segment element. You can also impose forces and moments on target elements. See [TARGE170](#) in the *Theory Reference for the Mechanical APDL and Mechanical Applications* for more details about this element. To represent 2-D target surfaces, use [TARGE169](#), a 2-D target segment element.

For rigid target surfaces, these elements can easily model complex target shapes. For flexible targets, these elements will overlay the solid, shell, or line elements describing the boundary of the deformable target body.

